AC 2012-5358: WORK-IN-PROGRESS: INTEGRATION OF HANDS-ON COMPUTATIONAL FLUID DYNAMICS (CFD) IN UNDERGRADUATE CURRICULUM

Dr. Yogendra M. Panta, Youngstown State University

Yogen Panta is an Assistant Professor of mechanical engineering at Youngstown State University, Ohio. He has been teaching and developing courses and research projects in the fluid thermal area. He is currently conducting applied research in thermo-fluids and computational fluid dynamics with local industries and federal agencies. Panta received a B.E. degree from Tribhuvan University, an M.S. degree from Youngstown State University, and a Ph.D. degree from the University of Nevada, Las Vegas. Panta’s research interests are in fluid dynamics, computational fluid dynamics (CFD), microfluidics/lab on chip, and energy research.

Dr. Hyun W. Kim, Youngstown State University

Hyun W. Kim is a professor of mechanical engineering in the Department of Mechanical and Industrial Engineering at Youngstown State University. He has been teaching and developing the Thermal Fluid Applications course and the companion laboratory course for the past few years. He is a registered Professional Engineer in Ohio and is currently conducting applied research in hydraulics and micro gas turbines. He helps the local industry and engineers with his expertise in heat transfer and thermal sciences. Kim received a B.S.E. degree from Seoul National University, a M.S.E. from the University of Michigan and a Ph.D. from the University of Toledo.

Param C Adhikari, Youngstown State University

Param Adhikari is currently working at Youngstown State University as a Graduate Research Assistant. He received a B.E. degree in mechanical engineering from Tribhuvan University. He is currently writing his master’s thesis on ”Analysis of Fluid Flow in Electro-osmotic Pumps.” His research interest is in fluid dynamics/modeling of fluid flow.

Mr. Sanket Aryal, Youngstown State University

Sanket Aryal is currently working at Youngstown State University as a Graduate Research Assistant. He received a B.E. degree in mechanical engineering from Kathmandu University. He is currently writing his master’s thesis on ”Analysis of Electro-osmotic Flow in DNA Chips.” His research interest is in microfluidics/lab on chips.
Work in Progress: Integration of Hands-on Computational Fluid Dynamics (CFD) in Undergraduate Curriculum

Abstract

Applied research facilities, such as computational fluid dynamics (CFD), wind tunnel testing and other experimental fluid mechanics facilities, will bolster students’ knowledge and undergraduate level research. Recent advancements in computational modeling/simulation and user-friendly graphical user interface of CFD code enable undergraduate engineering students to perform CFD analysis of heat and fluid flow problems providing better understanding of heat and fluid properties, and their phenomenon. Using CFD simulation tool in undergraduate research can significantly improve the understanding of various fluid flow phenomena as students are able to visualize the flow domains using the simulation for different boundary conditions. We describe an innovative plan for the development, implementation, and evaluation of an effective curriculum of CFD intended as an elective course for undergraduate and introductory course for graduate level students. The curriculum includes learning objectives, applications, conditions, exercise notes with a proposed course syllabus. One of the main objectives is to teach students from novice to expert users preparing them with adequate fluid mechanics fundamentals and hands-on CFD project works to prepare for their capstone design projects, higher education and advanced research in fluid mechanics. We have planned to incorporate a CFD educational interface for hands-on student experience in fluid mechanics, which reflects real-world engineering applications used in companies, government research labs, and higher education research.

1. Introduction

Computational fluid dynamics (CFD) has been included as a senior-level Thermal-Fluids Engineering course in the curriculum of mechanical engineering program at many US universities. In some universities, this course is adopted in the junior level undergraduate where the course structures for CFD are ranging from beginner, intermediate to advanced levels in other institutions. Thus, CFD is a widely adopted proven course that includes integration and differentiation of fundamental governing differential equations in fluid dynamics. CFD tool brings discretized algebraic forms to be solved for the flow field properties at discrete points over run time and/or coordinates space\. With the rapid advancement in the computers and their computing power, CFD has become more reliable, efficient, and an essential tool in the design, analysis and optimization of fluid flow devices for various engineering applications. Some examples of CFD application include the design of a new wind guard for a rooftop solar panel against high wind loads, design of subway tunnels, design of cooling systems for densely packed electronic enclosures, helping surgeons to understand the fluid flow in human body for clinical purposes, and optimizing the performance of a number of home appliances $^{1,2}$.

Due to the versatility, efficiency, credibility, and increasing use of modeling and simulation techniques in optimal design purposes for various fluid devices in industrial applications, CFD is becoming as a design and analysis tool in fluid power, thermo-fluid, oil, chemical processing, biotechnology, hydraulics and energy companies that make educators convince the need for
incorporating CFD course in the curriculum of undergraduate engineering education. Thus, number of universities that develops and implements a CFD course for undergraduate and graduate engineering students is growing every year, especially for mechanical, civil, biomedical, energy, and aerospace engineering disciplines. Incorporating a CFD into the fluid curriculum will not only benefit to have better understanding and visualization of fundamental fluid dynamics and prepare them for higher studies and research but also support to achieve their short and long term career goals. Furthermore, it is felt that an early introduction to CFD may inspire the students to take advanced fluid mechanics courses or go to graduate school.

The CFD course contents includes general fluid mechanics topics such as fluid properties, fluid statics, continuity equation, momentum balance, energy balance, internal and external flow, incompressible flow, inviscid flow, head losses in pipe flow, and lift/drag characteristics etc. Demonstrations supplement the lectures by providing students an opportunity to see first-hand experience for various aspects of fluid flow and the properties. This paper presents the design and implementation of a senior undergraduate level or a junior graduate level (sometimes called a swing course) course on CFD mainly intended for the department of mechanical engineering programs. One of the unique features of the course is the use of commercial industry-leading CFD software such as ANSYS Fluent, ANSYS, Inc. as the hands-on teaching/learning tool for the faculty/students. So, this paper reports the need for developing and implementing a CFD course in undergraduate engineering curriculum to introduce students, a modern tool, and equip them with necessary skills for becoming a better future engineer. This paper also includes an introduction of CFD methodology, CFD software, some sample projects, minimum computing resources and assessment tools required to develop and implement a course on CFD. Assessment tools include collecting students' feedback and course evaluation regularly. This will serve a baseline data to compare with the program objectives and the ABET outcomes.

2. Relationship of the CFD course with the educational objectives of the program and ABET criterion

As mentioned earlier, CFD course in the mechanical engineering program will strengthen to achieve the program objectives. A typical mechanical engineering program's objectives are to:

- provide an educational environment rich in opportunities for students to obtain the knowledge and skills that will prepare its graduates for successful careers as a mechanical engineer or for advanced studies.
- provide a comprehensive education for students capable to identify, formulate, and solve engineering problems by applying fundamental knowledge of mathematics, basic and engineering sciences, and by utilizing modern techniques, methods, skills, and tools.
- provide a strong engineering education for students to be able to design a system, components, or process to meet the desired needs, as well as to design and conduct experiments, and to analyze the acquired data and interpret the results.

The proposed CFD course in the mechanical engineering program will provide a general education, a complementary to its engineering education. Students will be able to:

- collaborate with multidisciplinary teams while working for their projects.
- communicate effectively while presenting their project results.
• understand the impact of engineering in the professional societies and fields through hands on computer projects that are directly related with real world applications.

More specifically, students are expected to have certain computational skills and be able to numerically analyze general engineering problems upon the completion of CFD course as expected by "specific outcomes of the course" and "student outcomes in ABET criterion" 6.

Specific outcomes of the course:
Upon completion of this course, students will be able to:
• build geometry, mesh that geometry, set up a CFD calculation on the mesh, perform the calculation, and post-process the results
• test a numerical result by comparison with experimental and analytical results
• qualitatively validate computational results with the physical principles of fluid mechanics

Student outcomes in ABET Criterion 6,7:
(a) strongly supported: Upon completion of this course, students will have:
• an ability to apply knowledge of mathematics, science, and engineering [Outcome (a)]
• an ability to use the techniques, skills, and modern engineering tools necessary for engineering practice [Outcome (k)]
(b) supported: Upon completion of this course, students will have:
• an ability to design and conduct experiments, as well as to analyze and interpret data [Outcome (b)]
• an ability to function on multidisciplinary teams [Outcome (d)]
• an ability to communicate effectively [Outcome (g)]
• an ability to identify, formulate, and solve engineering problems [Outcome (e)]

3. Introduction to Computational Analysis

Thus, the proposed CFD course is closely tied with program educational objectives and ABET outcomes as explained earlier. In this section, a brief introduction and methodology of CFD is presented in order to shed light on ANSYS Fluent, a commercially available CFD software.

Computational analysis involves solving a problem through the use of an algorithm and mathematical model. ANSYS Fluent allows the user to solve varieties of heat and fluid flow problems of 1D, 2D and 3D fluid flows, and combined heat/fluid flow to a highly complex nature of fluid flow problems. The software is built to model and analyze many types of laminar and turbulent fluid flows. The software has numerous features and add-ons that allow the user to model simple to complex geometries using selected mathematical models and numerical schemes 8.

Main components of ANSYS-Fluent software 8 are ANSYS Design Modeler, ANSYS-Meshing, the pre-processors, ANSYS- Fluent, the solver, and CFD-Post, the post-processor. Each component of the software has its own Graphical User Interface (GUI) to facilitate the user. The manual, help file and tutorials of the software are in electronic form and are installed locally in each workstation. Some of the tutorials provided with the software, such as static mixer and flow
from a circular vent, will be used to introduce the software to the students. These tutorials are extremely helpful to the students as they show the user step-by-step instruction to follow with occasional snapshots of the ANSYS-Fluent screens.

As mentioned earlier, the software has a workbench called ANSYS Workbench that comes with ANSYS Design Modeler for geometric modeling. A third party computer aided design (CAD) program, such as SolidWorks, can also be used to create the model geometry instead of ANSYS Design Modeler. The remaining add-ons include ANSYS Meshing for preprocessing and ANSYS Fluent for model setup, simulation and results presentation. The meshing software, depending on the quality of the mesh, enables the user to achieve varying degrees of accuracy; usually the finer the mesh, the more accurate the solution. The processing and post-processing of the meshed model were performed in ANSYS Fluent and visualization by CFD-Post. A brief introduction to CFD methodology and CFD solver techniques follow below.

**CFD methodology**

The procedure to set-up and run a successful simulation in ANSYS Fluent, for a fluid flow problem, consists of a series of steps that are completed sequentially as outlined below and shown in Figure 1.

a. **Construction of the geometrical models** using ANSYS Design Modeler or in another CAD program such as SolidWorks, or AutoCAD.

b. **Meshing of the fluid domain and setting up boundaries** of the geometrical model into discrete volumes using appropriate meshing parameters and techniques via ANSYS Meshing. It is advantageous to have smaller volumes near the points of interest of the model and areas where the physical phenomena of the fluid will be more prevalent and important.

c. **Determination and selection of the appropriate modeling technique** available in ANSYS Fluent that best conforms to the conditions and phenomena of the flow situation of the problem.

d. **Defining the boundary conditions and fluid properties**.

e. **Using the chosen solver in Fluent** iterate for converged solutions of continuity, momentum, energy and turbulence.

---

**Figure 1** CFD methodology

![CFD methodology diagram]
Figure 1 displays the methodology described above, in the form of a flow chart. **Pre-processing** is the first step that originates with the geometric creation/design of the model. This can be done one of the two ways; either by creating the geometry in **ANSYS DesignModeler** or by using a CAD program such as SolidWorks or AutoCAD. Once the model/geometry was constructed, it is necessary to develop the enclosure, or “flow field” of the model. After the wall boundaries, geometries, and enclosed domains were defined, **ANSYS Meshing** is followed next.

**ANSYS Meshing** allows the user to choose and apply various meshing schemes and techniques in order to discretize the flow domain for accurate simulation results. The meshing program discretizes the geometries, i.e. the flow field box, into small cells or volumes depending on whether the model is in 2-D or in 3-D analysis. The meshing applied to the flow field can be chosen from structured or unstructured mesh elements such as quadrilateral, triangular or quadrilateral plus triangular elements. Available mesh schemes include mapped mesh, edge sizing, element sizing, inflation, etc.

The **ANSYS post-processor** allows the user to evaluate, visualize, read and save the results obtained from the solver, qualitatively and quantitatively. An optimal meshing technique, in conjunction with a fixed size function, enables the user to attain very small elements on the surface and area surrounding the geometries that are within the mesh quality requirements in **ANSYS Fluent**. The extent of which the domain cells of the geometry are skewed, quantitatively less than 0.25 is considered a good meshing. **ANSYS Fluent** Manual guides that the orthogonal quality of mesh cells greater than 0.75 is considered a good meshing. Once the model is finely and properly meshed, it is then possible for the model to be exported to **ANSYS Fluent** for solution. **ANSYS CFD Post** allows the user to better visualize the simulation results through vectors, streamlines, plots, contours, animations etc.

4. **Course structure: Computational Fluid Dynamics (CFD)**

Thus, the proposed CFD course will be structured with a significant portion of CFD tutorial and exercise materials as introduced earlier. In this section, a brief introduction of CFD course structure is presented in order to shed light on course prerequisites, course description, course textbook, semester projects, and course evaluations.

The course is proposed to develop as an elective course for senior undergraduate students, however, first-year graduate students are also allowed to take the course. To be eligible for registration of the course and ultimately be able to understand the course materials, the students must have pre-knowledge of fluid mechanics, basic heat and fluid flow, partial differential equations, numerical methods, and preferably a programming language or a software package such as Matlab, MathCAD or C programming.

**Course prerequisites:**

The prerequisites for the CFD course are the completion of an undergraduate fluid mechanics and heat transfer course, an advanced calculus course, and a numerical analysis course. A typical curriculum of a mechanical engineering program and hierarchies of the prerequisite courses for the CFD course are shown in **Exhibit A** and **Exhibit B**. (summarized in **Table 1**).
Table 1 Hierarchies of the prerequisite courses of CFD course for a typical four-year undergraduate program in mechanical engineering (See details in Exhibit A and Exhibit B).

<table>
<thead>
<tr>
<th>Year/Semester</th>
<th>Fall</th>
<th>Spring</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>MATH 1571 Calc 1 CHEM 1515 Chemistry 1</td>
<td>MATH 1572 Calc 2 ENGR 1560 Engineering Computing</td>
</tr>
<tr>
<td>2</td>
<td>MECH2603 Thermodynamics 1 MATH 2673 Calc 3</td>
<td>MECH2604 Thermodynamics 2 MATH3705 Diff. Equations</td>
</tr>
<tr>
<td>3</td>
<td>MECH3720 Fluid Dynamics</td>
<td>MECH3708 Dynamic Systems Modeling MECH3720L Fluid Dynamics Lab MECH3725 Heat Transfer</td>
</tr>
<tr>
<td>4</td>
<td>MECH4835 Thermo-Fluid MECH4835L Thermo-Fluid Lab</td>
<td>MECH 5885 Computational Fluid Dynamics (CFD)</td>
</tr>
</tbody>
</table>

Course objectives

The class is designed with a 3 hours of lecture each week including some demonstration and practice labs for CFD simulation. A typical semester lasts for about 15 weeks that makes the total instruction to about 45 hours. As explained in the course description, main objectives of the course are:

(a) To develop understanding of physics and their mathematical formulations underlying CFD

(b) To provide students with hands-on experience using ANSYS Fluent, a commercial CFD

More specifically, course will introduce and explain the principles and methodology of CFD analysis and a CFD software tool, with simple examples, and will provide a basic understanding of general CFD procedures. Included in the course are the mathematical and physical fundamentals of CFD, formulation of CFD problems, basic principles of numerical approximation (grids, consistency, convergence, stability, and order of approximation, etc.), methods of discretization with focus on finite volume technique, methods of solution of transient and steady state problems, commonly used numerical methods for heat transfer and fluid flows, and a brief introduction into turbulence modeling.

The curriculum for the CFD course is developed to achieve the main objectives satisfactorily. The course is basically divided into three main parts: the first part of the course covers the physics and the mathematical foundations of CFD and the second part covers the CFD methodology to solve simple, intermediate and some advanced (only for graduate students) fluid flow problems. In the third part of the course, students will be assigned a computational fluid project. The first part of the course includes reviews of the theories and governing equations of fluid dynamics for CFD processing, discretization, grid generation, and solution methods. Even though majority of the problems for undergraduate students will include laminar and steady flow for incompressible fluid, an introduction to compressible fluid, turbulent flows and turbulent models will also be introduced. The second part of the course covers the use of ANSYS Fluent to solve fluid flow problems such as flow in a straight, converging, diverging, elbow, converging-diverging sections of pipe. Similarly, flow over a vertical plate, horizontal plate,
standard objects such as cylinder, sphere, triangle, rectangular blocks will be introduced to simulate fluid properties such as velocity, pressure, lift and drag. In the third part of the course, students will be assigned a computational fluid project as their semester projects using the ANSYS-Fluent software.

**Course description**

The definition of CFD and its applications in many areas of engineering, components of a typical ANSYS-Fluent - preprocessor, solver, and post-processor - and their functions will be discussed as an overview of CFD methodology. The case studies to solve some tutorial problems taken from ANSYS-Fluent tutorial materials will be incorporated in the course materials.

As mentioned earlier, first, physics and the governing equations of fluid dynamics and thermodynamics will be reviewed. The partial differential equations (PDE) for the conservation of mass, momentum, and energy will be derived in the class and the physical meanings of each term in the equations will be explained. Both differential and finite volume forms of the governing equations and their applications will be covered. The classroom discussions will be emphasized on the differences between diffusive, convective and source terms in the governing PDEs and the effects of these terms on the solution procedures of the PDEs. The general behavior of the PDEs and their solution methods will be discussed based on the classifications of the governing equations as elliptic, parabolic, or hyperbolic equation. The importance of boundary and initial conditions in solving each type of the PDE equations will also be presented.

Various types of discretization methods; finite difference, finite volume, and polynomial methods, will be discussed in class with examples. The problem associated with the convective term using the central difference, forward difference, and backward difference will be presented. The accuracy and errors of each method, convergence, stability, conservativeness, and boundedness will be covered. Materials from references will also be used as only the textbook coverage will not be sufficient for the topics outlined in the syllabus. Finite volume method will be emphasized and will be compared with other methods such as finite difference, and finite element in the class discussions. As the finite volume method (FVM) is the most common method used in the commercial CFD software such as ANSYS-Fluent, FVM will be described more detail in the class. Then, grid generation will be discussed for its different types and methods with particular emphasis on the body-fitted coordinate method and the adaptive grid method for their widespread uses in commercial software. The solution algorithms and numerical methods used for solving the discretized equations will also be presented. Common pressure-velocity solution algorithms such as SIMPLE (Semi-Implicit Method for Pressure Linked Equations algorithm by Patankar and Spalding (1972) and PISO (Pressure-Implicit with Splitting of Operators algorithm by Issa (1986)) will be covered with other solution algorithms and contrasted with each another to facilitate students’ understanding. Numerical solution methods covering both direct and iterative solution methods will be introduced. The direct methods - Gauss elimination and Thomas algorithm – and indirect methods – Jacobi and Gauss Seidel iteration methods will be covered.

Discussion on the explicit and implicit solution algorithms will also be presented in class. The turbulent flow and turbulent models will be discussed. The materials for the turbulent flow will
be taken from Versteeg and Malalasekera\textsuperscript{9}. In addition, some simple solutions from the text will be discussed and compared with the solution obtained from the ANSYS-Fluent software.

Following is a brief list of topics that will be covered in the class.

**Brief list of topics to be covered**

- Categorization of methods in science and engineering: experimental (physical); analytical; and numerical (e.g. CFD)
- Comparison of exact analytical solutions of differential equations to approximate numerical solutions
- Commercial CFD code – ANSYS Fluent
- The Static mixer problem
- Pipe flow: laminar and turbulent
- Laminar Flat Plate Boundary Layer: grid resolution issues
- Cylinder in a cross flow at a variety of Reynolds numbers

**Course textbook**

There are a number of textbooks on CFD available. Computational Fluid Dynamics: A Practical Approach authored by Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu is chosen as the textbook\textsuperscript{10}. The contents of this book are better suited to the undergraduate students as it explains the review of physics and mathematical foundations for fluid flow and the modeling tools using the CFD. Another book Versteeg and Malalasekera\textsuperscript{9} can also be used as a supplemental text. The chapter on turbulence and its modeling in Versteeg and Malalasekera\textsuperscript{9} can be used to cover the topic of turbulent flows in class. The instructor can also utilize other books on CFD\textsuperscript{11,15,16} and on numerical analysis for the course materials\textsuperscript{17}. *Exhibit C* includes a full list of references as outlined in the course syllabus.

**Course work and semester projects**

The inclusion of computational fluid dynamics projects in engineering provides the cooperative learning environment for students to apply theoretical knowledge and gain teamwork skills. A good beginning for the students to learn CFD simulation would be to follow the tutorial resources prepared by SimCafe-Cornell University\textsuperscript{18}. These tutorial problems for Fluent learning modules with ANSYS Fluent simulation are excellent sources for beginners to be ready for semester projects. Some of the tutorial problems include\textsuperscript{18}; (a) internal fluid flow such as laminar pipe flow, turbulent pipe flow, flow through an orifice, flow in a nozzle (details shown below) etc., (b) external fluid flow such as flow over a flat plate, steady flow over a cylinder, unsteady flow over a cylinder, flow over an airfoil, (c) heat and fluid flow such as forced convection over a flat plate, heat and fluid flow in a heater etc.

Each tutorial begins with a problem specification and a solution can be obtained by\textsuperscript{18}:

1. Pre-analysis and start-up
2. Create Geometry
3. Mesh Geometry
4. Set Boundary Types
5. Set Up Physics of the Problem
6. Solve
7. Analyze Results
8. Refine Mesh
9. Verification and Validation

Students will prepare a technical report and make a classroom presentation on the report for each project. There will be two projects in the course; first project serves to familiarize the students with the software and the second project will be the main semester project. The first project will be based on one of many validation problems used for CFD such as flow in a simple pipe, flow over a vertical plate, flow over a horizontal plate, flow on a cylinder etc. Students will also have to compare their computed results with the experimental results taken from the literature to validate their solution. The main purpose of the first project is to provide students with the ability to solve simple flow problems using ANSYS-Fluent software. In addition, the first project will provide students the knowledge to conduct validation and testing of the CFD software. For the second project which is also the semester project, each team will have to choose a practical engineering problem involving fluid flow and solve the problem using the software. Some of the sample problems include flow analysis in a high pressure pump, flow analysis in a fluid mixer, flow analysis in a piping network, flow analysis of various other flow devices such as venturi-meter, blood flow analysis in arterial network etc. These projects will be assigned upon the students' interests; however, some of them may be student-driven. The students will then be required to find the corresponding experimental results from the literature for each problem in order to validate their computational results. As for the first project, each group of students is required to prepare and submit an engineering report and present their findings in-class.

Sample semester projects:
(a) Pipe flow analysis

The analysis of a horizontal pipe provides the students an opportunity to better understand the fundamentals of fluid flow in a closed conduit. As an example for this lab project, students input a uniform velocity profile at the inlet of a converging pipe and investigate how the “no-slip condition” and fluid viscosity causes the velocity profile to change until it becomes fully developed.

Students will be given both laminar and turbulent flow cases to analyze fluid flow and is required to include the following in their laboratory report.

- Velocity vector plots at the entrance and exit of the pipe.
- Velocity profile plots at various cross-sections to demonstrate flow development.
- Comparison of CFD results with approximate hydrodynamic entry length for both laminar and for turbulent flow.
- Static pressure contours.
- Comparison of head loss calculated using Darcy friction factor with CFD simulation.
- Comparison of laminar and turbulent flow characteristics.
- Effect of different Reynolds Numbers within a flow domain.

A sample project: Analysis of fluid flow in a pipe nozzle

As an example, given problem statement, goal of the project, physical schematics of fluid flow, assumptions, governing equations, solution and results are presented below. Students can further extend “Results” section with more visualization figures and description in their actual project reports.
**Problem statement/Given:** A typical nozzle as shown in Figure 2 with varying diameters 75 mm at the inlet and 25 mm at the outlet to operate an inlet pressure of 700 kPa (gauge) to atmosphere as the outlet pressure. Analyze the fluid flow using CFD simulations and validate by hand calculation. This problem is taken from Fox and McDonald's Fluid Mechanics textbook. This type of proof-of-concept problem can be assigned as the first project in the CFD lab module.

*Physical schematics of fluid flow*

![Fluid flow in a nozzle (dimensions are in mm)](image)

**Figure 2** Fluid flow in a nozzle (dimensions are in mm)\(^{16}\)

**Goal:**
Using Analytical and CFD-
(a) **Numerical Results:** Outlet Velocity, Volumetric Flow Rate
(b) **Flow Visualization:** Contours- Velocity and Pressure and Vectors- Velocity

**Assumptions:**
Assuming an incompressible, inviscid and steady flow in the nozzle and ignoring the gravity effects, this problem can be solved for outlet velocity and mass flow analytically and CFD simulations.

**Governing equations:**
Applying Bernoulli’s equation between the inlet (1) and outlet (2);
\[
\frac{p_1}{\rho} + \frac{V_1^2}{2} = \frac{p_2}{\rho} + \frac{V_2^2}{2}
\]

\text{---------------------------}(I)
Using \textit{continuity equation} for mass flow:

\[ Q = V_1 A_1 = V_1 \frac{\pi D^2}{4} = V_2 \frac{\pi d^2}{4} \quad \text{(II)} \]

So,

\[ V_1 = V_2 \left(\frac{d}{D}\right)^2 \quad \text{(III)} \]

Combining equation (III) and equation (I); we get:

\[ V_2^2 - V_2^2 \left(\frac{d}{D}\right)^4 = \frac{2(p_1 - p_2)}{\rho} \]

So,

\[ V_2 = \sqrt{\frac{2(p_1 - p_2)}{\rho \left[1 - \left(\frac{d}{D}\right)^4\right]} } \quad \text{(IV)} \]

\[ V_2 = \sqrt{\frac{2(p_1 - p_2)}{\rho \left[1 - \left(\frac{d}{D}\right)^4\right]} } \]

\[ \text{Action/Solution:} \]

Above equations (V) and (II) are used to calculate the outlet velocity and the volumetric flow rate. Following \textbf{Table 2} shows the results and the comparative study of analytical vs. CFD simulations for velocity and flow rate.

\textbf{Table 2 Comparative Results:} Analytical vs. CFD for fluid flow in nozzle

<table>
<thead>
<tr>
<th>Simulation</th>
<th>2D simulation</th>
<th>Comparative results</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model</td>
<td>Nozzle Dia. D = 75 mm d = 25 mm</td>
<td>Outlet Velocity (m/s)</td>
</tr>
<tr>
<td>Inlet boundary</td>
<td>Pressure, p₁ 700 kPa (gauge)</td>
<td>Analytical</td>
</tr>
<tr>
<td>Outlet boundary</td>
<td>Pressure, p₂ 0 (gauge)</td>
<td>37.6</td>
</tr>
<tr>
<td>Walls</td>
<td>No-slip</td>
<td></td>
</tr>
</tbody>
</table>

\textit{Flow visualization:}

CFD simulations and the results of fluid flow in the nozzle can be visualized as shown in the following \textbf{Figures 3, 4} and \textbf{5} for the velocity contours, pressure contours and velocity vectors.
**Figure 3** Velocity contours for fluid flow in the nozzle

**Figure 4** Pressure contours for fluid flow in the nozzle
Figure 5 Velocity vectors for fluid flow in the nozzle

(b) Calculation of Lift/Drag

The analysis of a number of standard objects such as square, triangle, circle, diamond etc. (Figure 6) provides the students an opportunity to better understand the fundamentals of fluid flow and understand the lift/drag coefficients for external fluid flow problems. For this lab project, students input a uniform velocity profile at the inlet in the fluid domain and investigate how inlet boundary conditions (such as velocity or pressure) effect on the drag for the objects.

(A)     (B)     (C)     (D)     (E)     (F)

Figure 6 Various standard objects for drag/lift study
Students will be given laminar and turbulent flow cases to analyze the problems and is required to include the following in their laboratory report.

- Velocity vectors and velocity profile contours
- Calculation of lift/drag Coefficients for various fluid velocities
- Validation of CFD results for both laminar and for turbulent flow.
- Static pressure contours and plot of the pipe
- Comparison of laminar and turbulent flow characteristics
- Effect of different Reynolds Number within a flow regime on the flow characteristics.

An excellent source to learn about airflow on an airfoil and lift/drag characteristics is NASA (National Aeronautics and Space Administration) website \(^{20}\).

**Capstone Design Projects:**

Introducing CFD modeling in the junior year gives the students an opportunity to use Fluent for design and research projects in their senior year. Also, the students are more apt to take the finite element course as their professional elective to learn about all aspects of CFD modeling and simulation for the purposes of design and analysis.

One current project performed by a group of senior undergraduate mechanical engineering students for their capstone project is to *analyze the fluid flow characteristics in a high pressure safety valve for determination of its failure cause*. This project was initially provided by a local company that manufactures varieties of safety valves and fittings \(^{21}\). Students are investigating the cause of the failure in the valve due to high pressure water flowing into it. A preliminary 2D-simulation using the ANSYS-Fluent software predicts the failure areas of interest in the valve which agrees pretty well with the actual failed part. Given the inlet & outlet boundary conditions and the geometry of the valve, the valve model was modeled and simulated by using ANSYS-Fluent. Upon the complete understanding of the flow characteristics inside the valve, redesign and remodel of the valve will be done and suggested for implementation in the new design to the company. Some snapshots taken from the ANSYS-Fluent simulations are shown below:
**Figure 7** Geometrical modeling of a high-pressure valve using ANSYS-DesignModeler

**Figure 8** Fluid domain meshing of the high-pressure valve using ANSYS-Meshing
Figure 9 Pressure contours in the high-pressure valve as given by ANSYS-Fluent Simulation

Figure 10 Velocity streamlines in a high-pressure valve as given by ANSYS-Fluent simulation
Course evaluations

The evaluation of students includes evaluating students through daily class work, weekly homework, biweekly quizzes, midterm exams, computer projects and a final exam. The duration of each midterm exam is 75 minutes of which each contributes 10% of the total grade. The first midterm exam will cover the topics of governing equations, PDEs, and discretization methods. The second midterm exam covers the finite volume method, grid generation, solution algorithms, and numerical methods. Final exam will cover CFD solution analysis including consistency, stability, convergence, accuracy, and efficiency. The commercial CFD Software recommended for the course projects is ANSYS-Fluent, developed by ANSYS, Inc.

5. Assessment and student survey

The tools used to assess the CFD course will comprise of lecture and laboratory modules. Instructor with the help of an assigned graduate assistant will come up with Course Appraisal Questionnaire which will be given to the students at the end of the semester for both lecture and laboratory assessments including informal student feedback, and the quality of the student’s lab reports. A student survey will be conducted on the lecture materials and software materials to evaluate the course and the software's suitability in improving the learning process of students. Students will be evaluated different features of the software such as Graphical User Interface (GUI) and ease of use. The questionnaire will be made specific to the CFD class and its laboratory modules.

The students will be evaluated on the basis of four criteria:
- a. Complete written reports for the assignments.
- b. Project Report/Presentations.
- c. Homework/Classwork/Quizzes.
- d. Midterm/Final examination.

6. Concluding remarks

This paper discussed the design and implementation of an undergraduate CFD course intended for an undergraduate program of mechanical engineering. Detailed descriptions of the course including the text, course contents, exams, and projects are presented. The opinions of the students on the commercial software used for the class will be surveyed and implemented the course improvements for the next course offering. We describe an innovative plan for the development, implementation, and evaluation of an effective curriculum of CFD intended as a common course for senior undergraduate and junior graduate level students. The curriculum is being designed with learning objectives, applications, conditions, and exercise notes. One of the main objectives is to teach students from novice to expert users preparing them with adequate fluid mechanics fundamentals and hands-on CFD project works in order to prepare them for their capstone design projects, higher education and further research in fluid mechanics. We have planned to incorporate a CFD educational interface for hands-on student experience in fluid mechanics, which reflects real-world engineering applications used in companies, government research labs, and higher education. This course is currently being implemented as a senior undergraduate/first year graduate level in mechanical engineering program at Youngstown State University. Therefore, detailed results, outcomes and discussions about the course will also be published in an engineering journal paper after the actual implementation of the CFD course.
Acknowledgements

One of the authors (YMP) would like to thank Salim Bux, a graduate student of mechanical engineering at YSU, for his incredible help for ANSYS Fluent simulation.

References:

17. Mullen, BJ, Weidner S, FLUENT Learning Modules, SimCafe-Cornell University, addressing [https://confluence.cornell.edu/display/SIMULATION/Home](https://confluence.cornell.edu/display/SIMULATION/Home) on 03/15/2012.

**Exhibit A: Curriculum Sheet-YSU**
Undergraduate Program in Mechanical Engineering

<table>
<thead>
<tr>
<th>Student's NAME __________________________________________</th>
<th>University BANNER ID __________________________</th>
</tr>
</thead>
</table>

### Fall Semester

<table>
<thead>
<tr>
<th>1st Year</th>
<th>Hours</th>
<th>Grade</th>
<th>2nd Year</th>
<th>Hours</th>
<th>Grade</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENGL 1550 Writing 1 (GER, W)¹</td>
<td>3</td>
<td></td>
<td>MECH 2603 Thermodynamics 1¹</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>MATH 1571 Calc 1 (GER, MA)¹</td>
<td>4</td>
<td></td>
<td>MATH 2673 Calculus 3</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>CHEM 1515 Chem 1 (GER, NS)¹</td>
<td>4</td>
<td></td>
<td>PHYS 2611 Phys 2 (GER, NS)</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>GER Elective ( AL)² ³</td>
<td>3</td>
<td></td>
<td>CEEN 2601 Statics</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>ENGR 1550 Engr Concepts</td>
<td>3</td>
<td></td>
<td>ECON 2610 Princ.I (GER, SI)</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td></td>
<td>17</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Spring Semester

<table>
<thead>
<tr>
<th>3rd Year</th>
<th>Hours</th>
<th>Grade</th>
<th>4th Year</th>
<th>Hours</th>
<th>Grade</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENGL 1551 Writing 2 (GER, W)</td>
<td>3</td>
<td></td>
<td>MECH 2604 Thermodynamics 2</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>MATH 1572 Calc 2 (GER, MA)</td>
<td>4</td>
<td></td>
<td>MECH 2606 Materials</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>PHYS 2610 Phys 1 (GER, NS)</td>
<td>4</td>
<td></td>
<td>MECH 2641 Dynamics³</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>CMST 1545 Speech (GER, O)</td>
<td>3</td>
<td></td>
<td>MATH 3705 Diff Equations</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>ENGR 1560 Engr Computing</td>
<td>3</td>
<td></td>
<td>CEEN 2602 Str of Material</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td>CEEN 2602L Str Mat Lab</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td>16</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Summary

<table>
<thead>
<tr>
<th>Writing (W) and Speech (O)</th>
<th>3 (9)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Art &amp; Literature (AL)</td>
<td>2 (6)</td>
</tr>
<tr>
<td>Soc &amp; Institution (SI)</td>
<td>3 (9)</td>
</tr>
<tr>
<td>Pers &amp; Soc Responsibility (PS)</td>
<td>2 (6)</td>
</tr>
<tr>
<td>Nat Science (NS)</td>
<td>3 (12)</td>
</tr>
<tr>
<td>Mathematics</td>
<td>5 (18)</td>
</tr>
<tr>
<td>ENGR &amp; Other Engr Programs</td>
<td>6 (16)</td>
</tr>
<tr>
<td>Mechanical Engr</td>
<td>23 (56)</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>47 (132)</strong></td>
</tr>
</tbody>
</table>

**MECH Electives:**

- MECH 4800 Special Topics
- MECH 4823 HVAC
- MECH 5811 Solar Engr
- MECH 5825 Heat Transfer 2
- MECH 5885 Computational Fluid Dynamics (CFD)
- MECH 5852 Stress 2
- MECH 5872 Acoustics
- MECH 5884 Finite Element Anal

**Footnotes:**

1. Requires a grade of "C" or better.
2. Additional GER information can be obtained through their website at www.ysu.edu/ger/.
3. Only GER information can be C/NC; up to 3 semester hours per term and 12 semester hours total.

SEE THE UNDERGRADUATE BULLETIN FOR ANY ADDITIONAL INFORMATION.
<table>
<thead>
<tr>
<th>COURSE</th>
<th>PREREQUISITE(S)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENGR 1550</td>
<td>Eligible to take MATH 1513 or higher level math course.</td>
</tr>
<tr>
<td>ENGR 1555</td>
<td>High school geometry or equivalent.</td>
</tr>
<tr>
<td>ENGR 1560</td>
<td>ENGR 1550, MATH 1571 or concurrent, high school drawing or ENGR 1555. (ENGR 1555 may be taken concurrently.)</td>
</tr>
<tr>
<td>ENGL 1550</td>
<td>Placement Test or ENGL 1539 or ENGL 1540.</td>
</tr>
<tr>
<td>ENGL 1551</td>
<td>ENGL 1550 or Placement Test.</td>
</tr>
<tr>
<td>CMST 1545</td>
<td>Placement in ENGL 1550 based on Composition &amp; Reading Test results, or completion of ENGL 1539 or ENGL 1540.</td>
</tr>
<tr>
<td>PHIL 2625</td>
<td>None.</td>
</tr>
<tr>
<td>ECON 2610</td>
<td>MATH 1501, or Level 3 or higher on Math Placement Test.</td>
</tr>
<tr>
<td>MATH 1571</td>
<td>Either 4 high school units of math (including trigonometry) and at least Level 7 on the Math Placement Test, or Math 1513.</td>
</tr>
<tr>
<td>MATH 1572</td>
<td>MATH 1517.</td>
</tr>
<tr>
<td>MATH 2673</td>
<td>MATH 1572.</td>
</tr>
<tr>
<td>MATH 3705</td>
<td>MATH 2673.</td>
</tr>
<tr>
<td>STAT 3743</td>
<td>MATH 1572 or MATH 1585H.</td>
</tr>
<tr>
<td>ISEN 3710</td>
<td>MATH 1571.</td>
</tr>
<tr>
<td>PHYS 2610</td>
<td>High school physics or PHYS 1501. Prereq. or concurrent: MATH 1571.</td>
</tr>
<tr>
<td>CHEM 1515</td>
<td>1 unit of high school chemistry; 3 units of high school algebra and geometry (or MATH 1513 or equiv.), CHEM 1501 or 1 unit of high school chemistry. CHEM 1515R concurrent, unless exempt by placement exam.</td>
</tr>
<tr>
<td>CEEN 2601</td>
<td>MATH 1572, PHYS 2610 (or concurrent).</td>
</tr>
<tr>
<td>CEEN 2602</td>
<td>CEEN 2601.</td>
</tr>
<tr>
<td>CEEN 2602L</td>
<td>CEEN 2602 (concurrent).</td>
</tr>
<tr>
<td>ECEN 2632</td>
<td>MATH 1572 (Prereq. or concurrent).</td>
</tr>
<tr>
<td>MECH 2603</td>
<td>MATH 1572.</td>
</tr>
<tr>
<td>MECH 2604</td>
<td>MECH 2603, CHEM 1515.</td>
</tr>
<tr>
<td>MECH 2606</td>
<td>MATH 1572.</td>
</tr>
<tr>
<td>MECH 2641</td>
<td>CEEN 2601.</td>
</tr>
<tr>
<td>MECH 3708</td>
<td>MATH 3705, ENGR 1560, MECH 2641, ECEN 2632.</td>
</tr>
<tr>
<td>MECH 3720</td>
<td>MECH 2604, MECH 2641, MATH 3705.</td>
</tr>
<tr>
<td>MECH 3720L</td>
<td>MECH 3720.</td>
</tr>
<tr>
<td>MECH 3723</td>
<td>MECH 2606 (or concurrent). Must take with MECH 3723L.</td>
</tr>
<tr>
<td>MECH 3723L</td>
<td>MECH 3723 (concurrent).</td>
</tr>
<tr>
<td>MECH 3725</td>
<td>MECH 3720.</td>
</tr>
<tr>
<td>MECH 3742</td>
<td>MECH 2641, ENGR 1560.</td>
</tr>
<tr>
<td>MECH 3751</td>
<td>MECH 2606, CEEN 2602. Must take with MECH 3751L.</td>
</tr>
<tr>
<td>MECH 3751L</td>
<td>MECH 3751 (concurrent).</td>
</tr>
<tr>
<td>MECH 3762</td>
<td>MECH 2641, MECH 3751. Must take with MECH 3762L.</td>
</tr>
<tr>
<td>MECH 3762L</td>
<td>MECH 3762 (concurrent).</td>
</tr>
<tr>
<td>MECH 4800</td>
<td>Junior standing or consent of instructor.</td>
</tr>
<tr>
<td>MECH 4808</td>
<td>MECH 3708, MECH 3725, MECH 3742, MECH 3762. Must take with MECH 4808L.</td>
</tr>
<tr>
<td>MECH 4808L</td>
<td>MECH 4808 (concurrent).</td>
</tr>
<tr>
<td>MECH 4809</td>
<td>MECH 4808. Must be taken at next offering after completing MECH 4808.</td>
</tr>
<tr>
<td>MECH 4823</td>
<td>MECH 3725.</td>
</tr>
<tr>
<td>MECH 4835</td>
<td>MECH 3708, MECH 3725. Must take with MECH 4835L.</td>
</tr>
<tr>
<td>MECH 4835L</td>
<td>MECH 4835 (concurrent).</td>
</tr>
<tr>
<td>MECH 4881</td>
<td>MECH 3708.</td>
</tr>
<tr>
<td>MECH 5811</td>
<td>PHYS 2611, MECH 3725, or consent of Chairperson.</td>
</tr>
<tr>
<td>MECH 5825</td>
<td>MECH 3708, MECH 3725.</td>
</tr>
<tr>
<td>MECH 5836</td>
<td>MECH 3725.</td>
</tr>
<tr>
<td>MECH 5852</td>
<td>MECH 3751, MECH 3751L, MATH 3705.</td>
</tr>
<tr>
<td>MECH 5872</td>
<td>MECH 3708.</td>
</tr>
<tr>
<td>MECH 5884</td>
<td>MECH 3708, MECH 3725, MECH 3751.</td>
</tr>
<tr>
<td>MECH 5892 w/L</td>
<td>MECH 4881. Must take with MECH 5892L.</td>
</tr>
<tr>
<td>MECH 5885</td>
<td>MECH 3708, MECH 3720, MECH 3725.</td>
</tr>
</tbody>
</table>
Exhibit C: Proposed Syllabus for MECH 5885 Computational Fluid Dynamics/ 3 credit hours

Description

Applied Computational Fluid Dynamics (CFD) is an elective course in the senior undergraduate Curriculum in the Department of Mechanical Engineering. It provides an introduction to the use of commercial CFD codes to analyze heat and fluid flow problems of practical engineering interest. The course begins with a study of simple finite-difference and finite volume models of partial differential equations that contain important features of more complex CFD models. The SIMPLE model for solution to incompressible flow problems is described.

The major portion of the course involves using a commercial CFD code. Models of simple pipe geometries will be developed and studied for analyzing basic flow features and predict various fluid properties. Simple models will be simulated with initial and boundary conditions, and post-processing will be used for flow field visualization.

A brief overview of turbulence and turbulence modeling including the standard k-ε model to simulate turbulent flow fields will be introduced. At the end of the course students will understand the CFD methodology with hands on semester project about process of developing a geometrical model of the flow, applying appropriate boundary conditions, specifying solution parameters, and visualizing the results using ANSYS-Fluent software.

Time and Place

Days and time for the class and the lab

Instructor

Instructor name and address
Web site for the course:
Walk-in Office Hours:

Reading Material

Recommended Textbook

Lecture Notes
Copies of lecture notes will be made available. On-line documentation for ANSYS Fluent will be used. Supplemental references are listed on the last page.
Learning Objectives

- Preprocessing of a flow problem, identify all physical data necessary to set up and solve the various fluid properties such as velocity, pressure, and temperature fields using a CFD package.
- Use ANSYS Fluent to solve laminar and simple turbulent flow problems.
- Identify whether and when a run of ANSYS-Fluent has converged.
- Describe the key features of a flow field represented by contour and vector plot of the velocities, and a contour plot of the pressure field.

Evaluation

Learning of the course material will be evaluated by grading of homework, quizzes, midterm exams, a semester project and a final exam. The midterm may have an in-class component and a take-home component. The final exam will be comprehensive. All exams are mandatory. Quizzes will be objective and will be administered biweekly. There will be two projects; first one is intended to make students familiarize with the software and the second is the semester project that carries 20% points. The cumulative score is based on the following weights:

<table>
<thead>
<tr>
<th>Percentage</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>20%</td>
<td>Homework</td>
</tr>
<tr>
<td>10%</td>
<td>Quizzes</td>
</tr>
<tr>
<td>20%</td>
<td>Midterm</td>
</tr>
<tr>
<td>25%</td>
<td>Projects (Two- second project is the semester project and carries 20%)</td>
</tr>
<tr>
<td>25%</td>
<td>Final Exam</td>
</tr>
</tbody>
</table>

Project

One quarter of the course grade will be based on two projects. One of them is the semester project, which involve a detailed simulation of a particular flow problem. Project topic may be chosen from a list of projects that have already been identified, or, with instructor approval, or you may come up on your own. Each student will have to submit a technical report and give a brief presentation on their project.

Supplemental References

References for CFD.


References for Fluid Mechanics.


